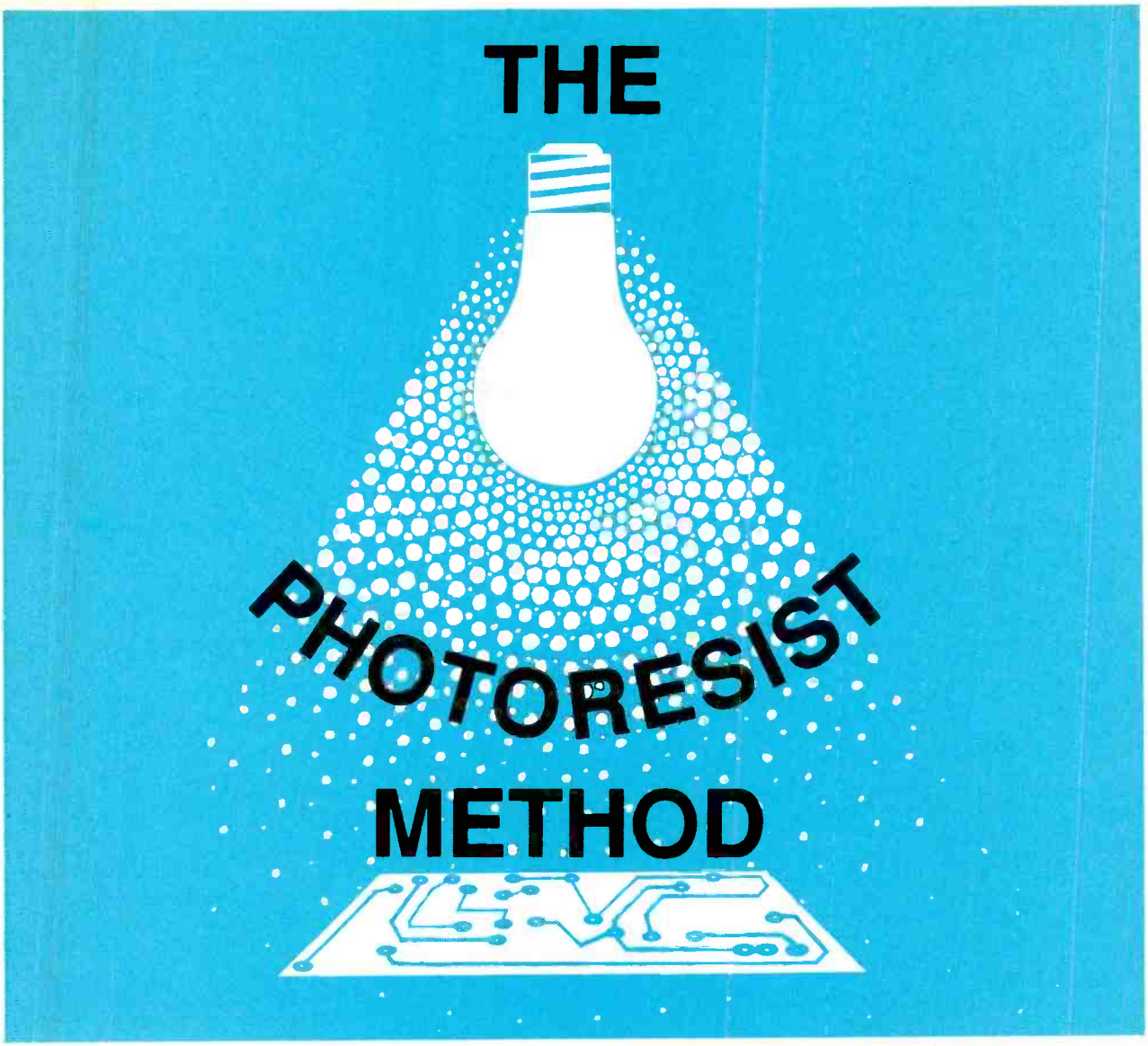
THE PHOTORESIST METHOD



Learn how to use the photo-resist method for making your own PC boards.

JOHN BERCIK

ANYONE CAN MAKE PROFESSIONAL single- and double-sided prototype circuit boards using just a few basic materials and techniques. The techniques shown in this article will allow you to produce a PC board for your prototype in a matter of hours. Board houses can often take weeks to produce a board. Once you are familiar with the techniques described in this article, you will expose, develop, and etch boards in about 40 minutes, at an average cost of about $25 each.

Boards produced with the photoresist method can have traces as thin as 15 mils (a mil is 1/1000 of an inch) with 10-mil isolations. Pads of 65 mils and drill holes of around 30 mils can also be produced. Once you get the hang of the procedure, you can make many copies of the same board with a sucess rate of around 95%; the 5% of the boards that are unusable usually result from reusing etchant and developer too much.

Many of the things you'll need to make PC boards using this technique can be found right in your kitchen, so you don't have to break the bank to try it out. Table 1 shows a list of the necessary equipment, and Figure 1 shows some of those materials.

Artwork

There are many sources for the artwork for your board layout. For you own designs, if you have access to a CAD program and a laser printer, you can get a clear plot from the program onto paper. If the circuit is small you should plot the artwork at double the actual size (a 2:1 scale). If you can't use a computer for your board design, you will have to produce your PC

www.americanradiohistory.com

TABLE 1—NEEDED MATERIALS

- Artwork (from a printer, preferably a laser printer)
- Sunlamp (not an infrared heat lamp)
- Darkroom safe light
- Kitchen timer
- A piece of plate glass larger than the desired board size (you can use the glass from a cheap picture frame)
- Hot plate
- Glass brownie pan
- Candy thermometer
- Plastic tray
- Bucket
- 1 pint of etchant
- 1 pint of developer
- Light sensitized PC-board material (single- or double-sided)
- Rubber gloves resistant to chemicals (available at hardware stores)
- Drill press and small bits (around .030- or .045-inch)

FIG. 1—HERE ARE SOME OF THE MATERIALS you need to make your own PC boards.

pattern on clear acetate film with black tape and donuts. Taping is recommended only for small boards. If CAD software is available don't use tape at all unless you are experienced in laying out boards by hand.

Next, you might need to have a film, or transparency, made of the artwork. (It is not necessary if your artwork is on acetate and at a 1:1 scale.) Printing shops charge about $15 a sheet. Two sheets are required for double-sided boards unless the patterns are small enough that they will both fit on one sheet (see Fig. 2). Try to keep a 1- to 2-inch border around each piece of artwork.

Determine which side of the transparency you want the emulsion (the image) on, so that when the film is laid on top of the PC-board blank, the pattern is touching the board and not on the side of the film facing away from the board. That prevents light from going under the artwork and causing traces to be etched away.

You must also specify if the film is to be scaled from the paper. If your paper plot is twice normal size, you'll have to get a 50% reduction to get the proper size. Finally, specify if you want a positive or negative. If your film is positive, use positive-sensitized boards and positive developer. With negative film, use negative-sensitized boards and negative developer.

Making a board

The following procedures are necessary to make double-sided boards; some of them are not necessary for single-sided boards. Once you have the film ready, cut a 2-inch border around the solder-side film and a 1-inch border around the component-side film.

Place the component-side film on top of the solder-side film with the correct orientation to simulate a finished board. Line up the patterns and tape the component-side film to the solder-side film along the top and side edges. Don't cover any of the artwork with tape. Insert a 1-inch wide scrap piece of PC-board material along the bottom edge to act as a spacer between the two films. Position the scrap material ¼-inch away from the edge of the artwork on the film. Then tape the material in place on both sides making sure that the through holes on the film still line up. Remove the tape from the sides and top of the film leaving only the scrap stock connecting the films.

You will need light-sensitized board material, developer, and etchant to complete the board. You can use positive or negative techniques as long as your film matches, although the positive technique is described here. Handle the sensitized boards by the edges so that the coating remains intact.

Place the sun lamp 10 inches

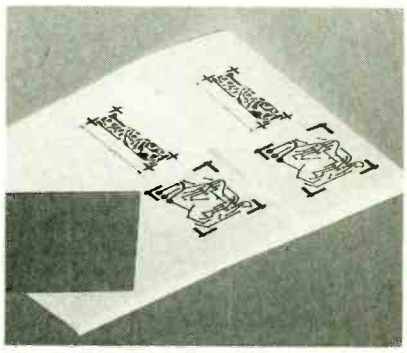


FIG. 2—PUT ALL PATTERNS on a single sheet of film, if possible, to save money on having the film made.

www.americanradiohistory.com

FIG. 3—ETCHANT IS AVAILABLE from many different suppliers, and in many different forms.

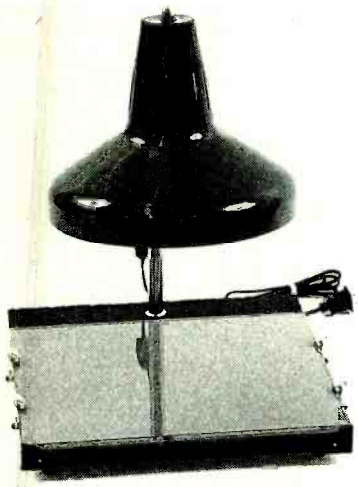


FIG. 4—A SUNLAMP IS MOUNTED to a base that makes exposing the blanks relatively easy. The base holds the glass securely in place.

above the surface to be exposed. Set up the hot plate with the glass brownie pan on it. Mount the candy thermometer to the side of the brownie pan. (The author softened a piece of scrap plastic into the shape of a clip and drilled a hole in it to hold the thermometer firmly).

The etching solution should be prepared before exposing your board, so that it's ready when you are. Ferric chloride can be used to etch the boards, although environmentally safer sodium persulfate etchant can also be used. Figure 3 shows different kinds of etchants. Remember, when working with caustic chemicals, make sure there's good ventilation, wear old clothes because the etchant stains don't wash out, and wear rubber gloves.

Place the glass pan on the hot plate and pour the etchant about 1½ inches deep into the pan. Turn on the heat and get the etchant stabilized to between 125 and 135 degrees Fahrenheit—do not let the etchant get over 140 degrees.

Fill the bucket with cold water. Get a clean glass bottle and mix the developer with the correct ratio of hot water (according to the manufacturer's directions—it might not be necessary at all), and pour it into the plastic developing tray.

Place your film, the large piece of glass, some tape, the sensitized board in its sealed, light-tight plastic bag, and a knife in front of you on a clean surface. Turn on the red safety light and turn off the other lights. Make sure the etchant is up to temperature. Open the sensitized board in the red light and, holding it by the edges, place it between the two films. Flatten the film out beneath a piece of paper (so you don't touch the board) and tape the film firmly to the board on both sides making sure not to upset the alignment. You can cut the corners off the film to make room for the tape.

Place the board and film under the sun lamp with the glass on top to keep the film pressed flat against the board (see Fig. 4). Set the timer for 6½ minutes and turn on the sun lamp being careful not to look directly at the bulb. After the board has been exposed for 6½ minutes, turn it over very quickly and reset the timer for another 6½ minutes. Finally, turn the sun lamp off. Wearing a rubber glove, remove the film and dip the board into the warm developer. Use a stick to move the board around in the developer. Lift it up and down from the edge to get a wave action going. Also flip it over a couple of times. A faint image will appear after a minute or two.

After about 3 or 4 minutes, take the board out and plunge it into the bucket of cold water. That stops the development process and hardens the etch resist.

This is a good stopping point if you plan to make multiple copies of the same board. as you can set the developed board aside, expose more of them, and then etch them all at once. If you are making only one board, you can turn on the lights at this time. After the board has been in the cold water for about a minute, remove it and then place the board into the etchant. After about 1 or 2 minutes, begin moving it around with a stick continuously to prevent dissolved copper from sticking to the board. Check that your temperature is correct as you continue etching. It should take from 5 to 15 minutes to etch, depending on how fresh the etchant and how big the board is. An etching tank—a tank specially made to etch and agitate PC boards—will speed up the process, and is a worthwhile investment should

www.americanradiohistory.com

you decide to make boards on a regular basis. You can buy an etching tank from suppliers that carry materials for making PC boards, or even build one (see **Radio-Electronics**, December 1989).

When you can see that the unwanted copper is almost fully etched, take the board out of the etchant and plunge it into the bucket of water to rinse off the etchant. If it is not etched enough, put it back into the etchant. It's better to stop etching too soon and then put it back in again than to etch it too much. When the board is done, rinse it off and towel dry it.

If your board does not etch completely, even with fresh developer and etchant, you must repeat the entire procedure, exposing the board a bit longer to the sun lamp. However, if traces are being etched away along with the unwanted copper, then you must reduce the exposure time. Once you determine the correct exposure time, it will work every time. Call your local city officials to find out how to properly dispose of the used chemicals—don't just pour them down the drain!

With a piece of fine steel wool, polish the board to remove all remaining etch resist. Inspect the board for any broken traces. You are now ready to drill the board. Remember to wear safety goggles. Although a drill press is best for drilling PC boards, a hand drill and a steady hand will also work.

When double-sided PC boards are professionally made, holes are plated through the board to connect traces from top to bottom. With our technique there is no plating through of the holes. We simulate the through-holes by inserting short lengths of wire through the holes and soldering them on both sides. In through-holes where component leads pass through, the leads are simply soldered on both sides. Because some components have leads that are difficult to solder on the component side, it is good practice to place as many traces as possible on the solder side when laying out the board.

When your board is complete, you can assemble your prototype. If everything checks out, you are ready to send the information to your PC board manufacturer and have them make as many boards as you need, with the assurance that they will work properly. Ω

SOURCES

Everything you need to make your own PC boards can be obtained from the following suppliers:

- Kepro Circuit Systems, 630 Axminister Drive, Fenton, MO 63026-2992 (800) 325-3878, (314) 343-1630
- Datak Corporation, 3117 Paterson Plank Road, North Bergen, NJ 07047 (201) 863-7667.
- A list of circuit-board houses who provide prototyping and full production services for boards up to 16 layers can be obtained free of charge from Skychaser, 980 Sherwood Place, Eugene, OR 97401 (503) 345-4609.

AUDIO SCRAMBLING

continued from page 42

the handset). The handset should preferably have an electret microphone. However, carbon microphones (found in older phones) can be used, if necessary, but R19 should be changed to about 1K, and R20 might have to be increased if excessive audio from a carbon microphone overdrives Q1, causing distortion.

Depending on the phone you have, you must make the proper jumper connections on the PC board near J1 and J2. A Radio Shack telephone model No. ET-171 (cat No. 43-374) was used in the author's prototype.

You must identify the following things on your phone(s):

1. The handset microphone and earpiece connections
2. The type of microphone (electret, dynamic, or carbon)
3. Microphone polarity (if it's the electret type)
4. The base connections

There are usually four wires that connect a telephone handset to its base. If you can't visually identify the wires after disassembling the handset, try connecting a 1.5-volt battery to alternate pairs of wires on the handset until you hear a click in the earpiece. Mark these as the receiver leads; there should be between 50 and 1000 ohms between them. The other two leads are for the microphone.

Check for short circuits between both of the receiver leads and the microphone leads with an ohmmeter on a high resistance range. A low resistance or a short between any two leads indicates that they are the ground leads for the microphone and receiver. If you find no continuity between the two sets of leads, connect the microphone pair to the test circuit shown in Fig. 9, and see if the microphone works; if not, reverse the connections. This will identify the microphone's hot and ground leads. Once you've identified all of the handset leads, note their positions on the modular connector. The telephone base connections can now be determined from the positions of the handset leads at the modular connector.

When you have all of the telephone connections identified, install the jumpers on the PC board near jacks J1 and J2. In Fig. 6 there are four jumper pads labeled A–D near each modular jack; the pads are also labeled by function. Once you know the signal positions at jacks J1 and J2 for your phone, install four jumpers per jack to properly route the signals.

The finished board can be mounted in a case like the one pictured in Fig. 8, or in any other suitable case. The case pictured allows the telephone to be placed on top of the scrambler without taking up any extra space.

With a pair of scrambler phones in hand, you're ready to start talking. All you need now is someone to talk to and a confidential topic to discuss. Ω

www.americanradiohistory.com